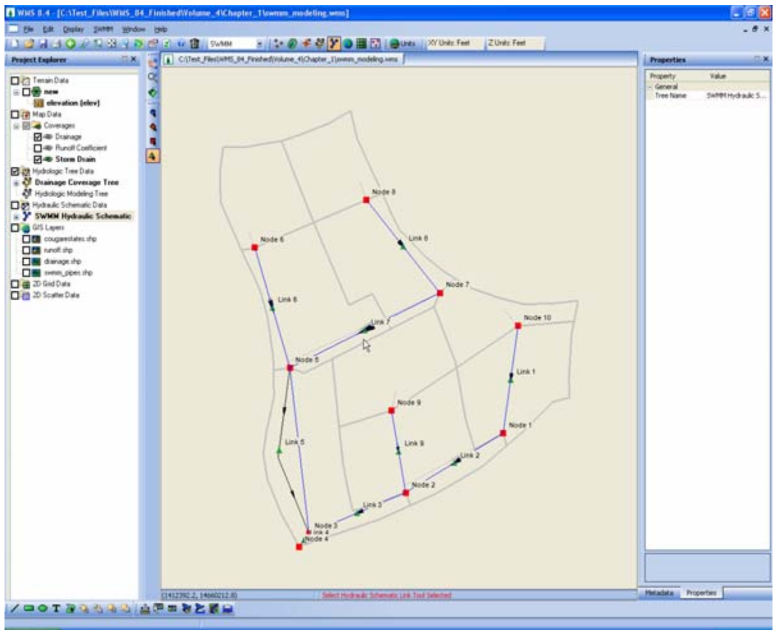
WMS

v. 8.4

WMS 8.4 Tutorial

Storm Drain Modeling – SWMM Modeling

Learn how to link a hydrologic model to the SWMM storm drain model



Objectives

Build a rational method hydrologic model and compute sub-basin flows. Import storm drain network information and link the storm drains to the hydrologic model. Run the flows from the hydrologic model through the storm drain model using either xpswmm or EPA-SWMM.

Prerequisite Tutorials

- Watershed Modeling – Rational Method Interface
- Editing Elevations – Using TINs

Required Components

- Data
- Drainage
- Map
- Hydrology
- Hydrologic Models
- Storm Drain

Time

- 30-60 minutes

AQUAVEO™

1 Contents

2 Introduction

SWMM is a hydrologic and hydraulic analysis tool used primarily designing and analyzing for storm drain systems. Hydrologic parameters are defined to obtain hydrographs and peak flows which can then be fed into the hydraulic component of the model. The hydraulic component lets engineers analyze the capacity of a current storm drain system or design a system to meet certain inflow conditions. WMS supports both EPA SWMM, developed by the Water Supply and Water Resources Division of the US Environmental Protection Agency's National Risk Management Research Laboratory, and xpswmm, developed by XP Software.

3 Objectives

In this exercise, we will set up a drainage simulation based on the Rational Method for a proposed subdivision. The objective of this exercise is to teach you the basic steps for defining a SWMM input file, running the numeric model, and viewing the results. These steps include the following:

1. Define runoff coefficients
2. Compute flows using the Rational Method

3. Import a pipe network
4. Run SWMM and view results
5. Route hydrographs using SWMM and view results

4 Defining Runoff Coefficients

4.1 Reading in the Subdivision Layout and Elevations

We will open a shapefile containing the geography of the proposed subdivision. Also, to help us in defining elevations and slopes for the drainage area, we will open a TIN (Triangulated Irregular Network) for the area:

1. Close all instances of WMS
2. Open WMS
3. Select ***File* | *Open***
4. Locate the folder ***C:\Program Files\WMS84\tutorial\stormrat***
5. Open *"cougarestates.tin"*
6. Right-click on the Coverages folder in the Project Explorer
7. Select ***New Coverage*** from the pop-up menu
8. Change the Coverage type to Runoff Coefficient
9. Select OK
10. Select ***File* | *Open***
11. Open *"cougarestates.shp"*
12. Switch to the *GIS* module
13. Select ***Mapping* | *Shapes->Feature Objects***
14. Select Yes to use all visible shapes
15. Select Next
16. Ensure that the Runoff coefficient is mapped
17. Select Next and then Finish
18. Hide *cougarestates.shp* by deselecting it in the Project Explorer

4.2 Defining the Runoff Coefficients

We will turn off the display of the TIN in order to better distinguish the subdivision polygons.

1. Toggle the check box for the "new" TIN off in the Project Explorer
2. Switch to the *Map* module
3. Choose the *Select Feature Polygon* tool

4. Use Figure 4-1 to assign runoff coefficients by double clicking each polygon and assigning the appropriate coefficient.

Figure 4-1: Assigning runoff coefficients.

5 Compute Flows from the Rational Method

Now we will set up our Rational Method analysis to compute peak flows for each subbasin. We will use the computed peak flows in our SWMM storm drain network.

5.1 Creating the Drainage Coverage

1. Select the Drainage coverage from the Project Explorer to make it the active coverage
2. Select ***File | Open***
3. Open *"runoff.shp"* and *"drainage.shp"*
4. Switch to the *GIS* module
5. Select ***Mapping | Shapes->Feature Objects***
6. Select Yes to use all visible shapes
7. Select Next, Next, Next, and then Finish

Figure 5-1 shows how the layout should look.

8. Hide *runoff.shp* and *drainage.shp* by deselecting their icons in the Project Explorer

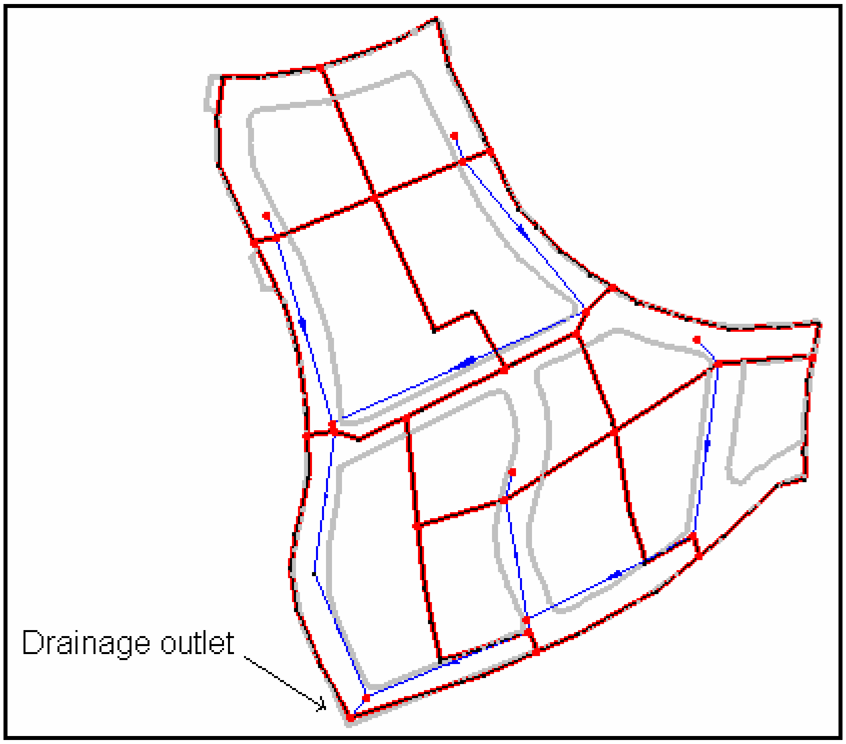


Figure 5-1: Drainage basins and streams for the subdivision.

Before defining nodes as storm drains/drainage outlets, we will hide the Runoff Coefficient coverage in order to simplify the screen display

9. Toggle the visibility check box for the Runoff Coefficient coverage off in the Project Explorer
10. Ensure that Drainage is the active coverage and that your current module is the *Map* module
11. Choose the *Select Feature Point/Node* tool
12. Double-click on the node labeled Drainage Outlet in Figure 5-1
13. Change the Point Type to Drainage Outlet and select OK
14. Select ***Feature Objects* | *Compute Basin Data***
15. Click the Current Coordinates button
16. Set the horizontal system to use a local projection and set the horizontal and vertical units as U.S. Survey Feet.
17. Select OK
18. Set Basin Areas to Acres and Distances to Feet
19. Select OK

5.2 Computing Runoff Coefficients

Composite runoff coefficients must be computed for each drainage area. A weighted average for each drainage area is calculated from data in the Runoff Coefficient coverage. To compute the basin runoff coefficients:

1. Switch to the *Hydrologic Modeling* module
2. Select ***Calculators | Compute GIS Attributes***
3. Set the Computation type as Runoff coefficients
4. Select OK

Composite runoff coefficients for each drainage area are computed and displayed on the screen.

5.3 Entering Times of Concentration

The Time of Concentration for a basin can be calculated using a Time Computation coverage. For this exercise, however, it is assumed that the TC values have already been calculated, and we will input them manually.

1. Change the Model drop-down box (located towards the top of the WMS interface) to Rational
2. Choose the *Select Basin* tool
3. Double-click on the basin icon for the basin in the lower left-hand corner of the subdivision
4. Enter 14 (minutes) for the Time of Concentration
5. Select each of the remaining drainage basins and assign Time of Concentration values using Figure 5-2 as a guide

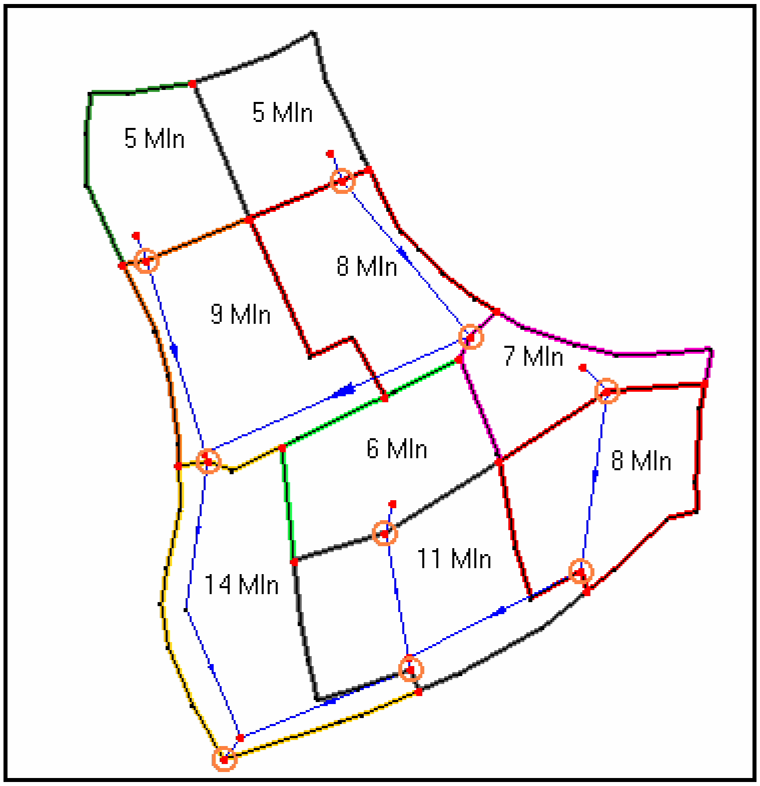


Figure 5-2: TC values for the drainage areas.

6. Select OK to close the Rational Method dialog

We have now set up a traditional rational simulation for the Cougar Estates subdivision. The only remaining step is to define the IDF curves.

5.4 Defining Rainfall Data

1. Double click on the basin icon for the basin in the lower left hand corner of the subdivision
2. Click on the Compute… button for the parameter labeled Compute I – IDF Curves
3. Choose the User Supplied Data option as the IDF curve computation
4. Click the Define Storm Data button
5. Change the Recurrence value to 25 yr
6. Enter the precipitation values shown in Figure 5-3

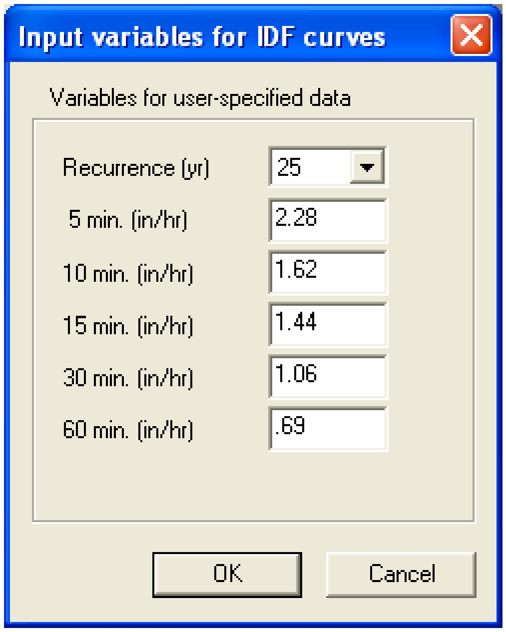


Figure 5-3: Values for computing the 25 yr IDF curve.

7. Select OK
8. Highlight the line corresponding to the 25-yr precipitation values from the window in the upper right-hand corner of the dialog
9. Click the *Compute Intensity* button.
10. Select Done
11. In the Display section, change the Type drop down menu to Outlets and change the Show drop down menu to All
12. Click the Compute… button next to the Compute I – IDF Curves parameter for each basin and outlet. Click the Compute Intensity button and click Done until each basin and outlet has a rainfall intensity displayed in the Parameters spreadsheet
13. Notice that a flow rate has now been computed for each basin and outlet. Click OK to exit the Rational Method dialog

We will now import a pre-defined storm drain network, assign properties to the network, and link it to the Drainage coverage.

6 Importing the Pipe Network

As mentioned above, we will be importing a pipe network to use in our SWMM model. This network was defined as a shapefile, and will be converted to feature objects in WMS. WMS can also open DXF files and convert them to feature objects. Alternatively, users can manually create a pipe network using the Create Feature Arc tool.

1. Right-click on the Coverages folder in the Project Explorer
2. Select ***New Coverage***
3. Change the Coverage type to Storm Drain

4. Select OK
5. Select ***File | Open***
6. Open *"swmm_pipes.shp"*
7. Switch to the *GIS* module
8. Select ***Mapping | Shapes->Feature Objects***
9. Select Yes to use all visible shapes
10. Select Next, Next, and Finish
11. Hide *swmm_pipes.shp*

6.1 Apply SWMM Attributes

1. Switch to the Map module
2. Select ***Storm Drain | Map->1D Schematic***

This will convert our storm drain coverage to a network of links and nodes recognized by SWMM. This command also automatically maps the elevations from our TIN to the ground elevations stored at each node. WMS has also computed the length of each storm drain link and applied a default invert elevation of eight feet below the ground elevation. Your project should now look similar to Figure 6-1.

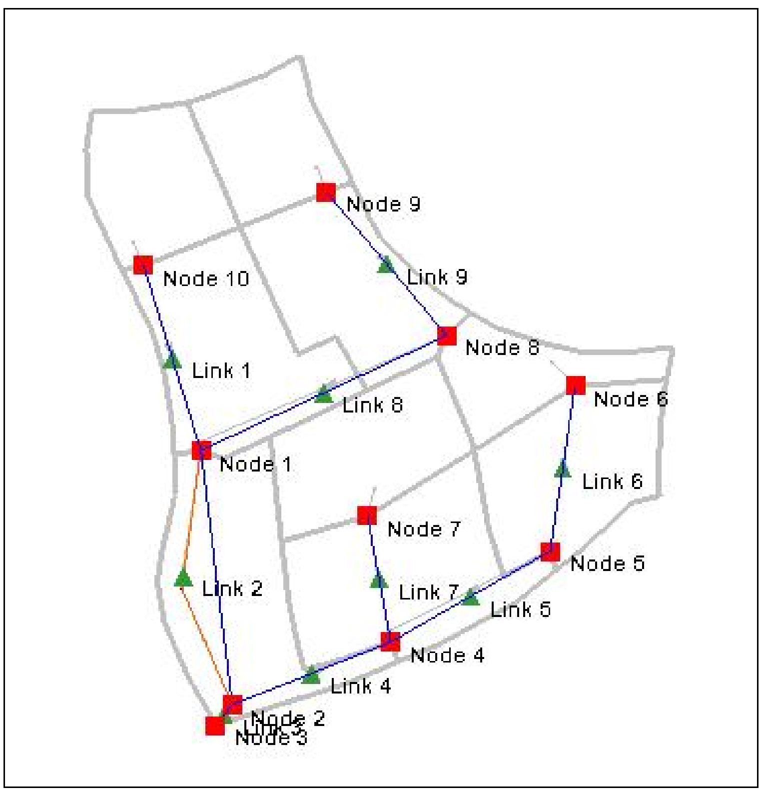


Figure 6-1 Link-Node Schematic for SWMM Model

3. In the Project Explorer, select the *SWMM Hydraulic Schematic* tree item

4. Choose the *Select Hydraulic Node* tool 
5. Double click on any node

In the node Properties dialog, notice the invert elevation of Node 10 and Node 1 (according to Figure 6-1). Storm water is supposed to flow from Node 10 to Node 1, but the default invert elevation is higher at Node 1 than at Node 10. We have a similar issue with respect to Node 9 and Node 8. This dialog lets us change the default invert elevations at each node.

Important Note: for the following steps, node numbers may not match node numbers in your model. Use Figure 6-1 as a reference to determine which nodes correspond to node and link names in your model.

6. Change the invert elevation of Node 1 to 4501.8

This elevation must be higher than that at Node 2. The elevation at Node 2 is 4501.5, so we've met this condition

7. Change the invert elevation of Node 9 to 4508.0

This elevation is now higher than the elevation at Node 8 (4507.5). Now notice the column labeled Linked Outlet Name. The storm drain nodes are not currently linked to our hydrologic analysis. We want to use the peak flows calculated in our Rational Method Analysis as inflows to our storm drain system.

8. Click OK on the node Properties dialog
9. Right click on the SWMM Hydraulic Schematic in the Project Explorer and select Link Outlets to Nodes
10. In the Link Storm Drain and Drainage Nodes dialog, click the Auto Link button.

This will automatically link the storm drain nodes to the drainage outlets that are within a given tolerance of the node. As you can see, Node 2 is not located close enough to any drainage outlet and so has not been linked

11. Click OK to exit the Link Storm Drain and Drainage Nodes dialog

Now that we've changed the elevations at our nodes and linked them to our outlets, we must also modify the upstream and downstream invert elevations of our links.

12. Choose the Select Hydraulic Link tool 
13. Double click on any link

The link Properties dialog lets us specify the dimensions of our storm drain pipes. We'll assume a circular shape for all pipes and no initial flow or depth. The lengths and invert elevations are shown, but the invert elevations at Link 1 and Link 9 need to be updated to reflect the changes we made to the nodes. We also need to define the diameter of the pipes. **Remember:** Use Figure 6-1 as a reference to determine which links correspond to link names in your model.

14. Change the Downstream Invert Elevation of Link 1 and the Upstream Invert Elevation of Link 2 to 4501.8
15. Change the Upstream Invert Elevation of Link 9 to 4508.0

16. In the Diameter/Height column of the link properties spreadsheet, enter a diameter of 4 feet for each link
17. Click OK to exit the link Properties dialog

Now we're ready to export a SWMM project file and run SWMM. Section 7 describes how to export and run an xpswmm model. To export and run an EPA SWMM model, skip to Section 8.

7 Running xpSWMM and Viewing Results

This section describes how to export and run an xpswmm model. The xpswmm model offers many features that can be used to modify your hydrologic analysis, customize your drain system, and analyze the model output. The features covered here represent just a few of the features you might find helpful.

1. Select ***SWMM | Run xpswmm…***
2. Name the file "*swmmtut.xpx*" and click OK

WMS saves an xpswmm project file and as long as xpswmm is loaded on your computer, it launches xpswmm and automatically opens the project we just created.

7.1 Running the SWMM Model

1. When xpswmm opens, if prompted click OK to convert the version 10.0 database and close the window displaying errors and warnings.
2. Your xpswmm window should now display an image similar to that shown in Figure 7-1.

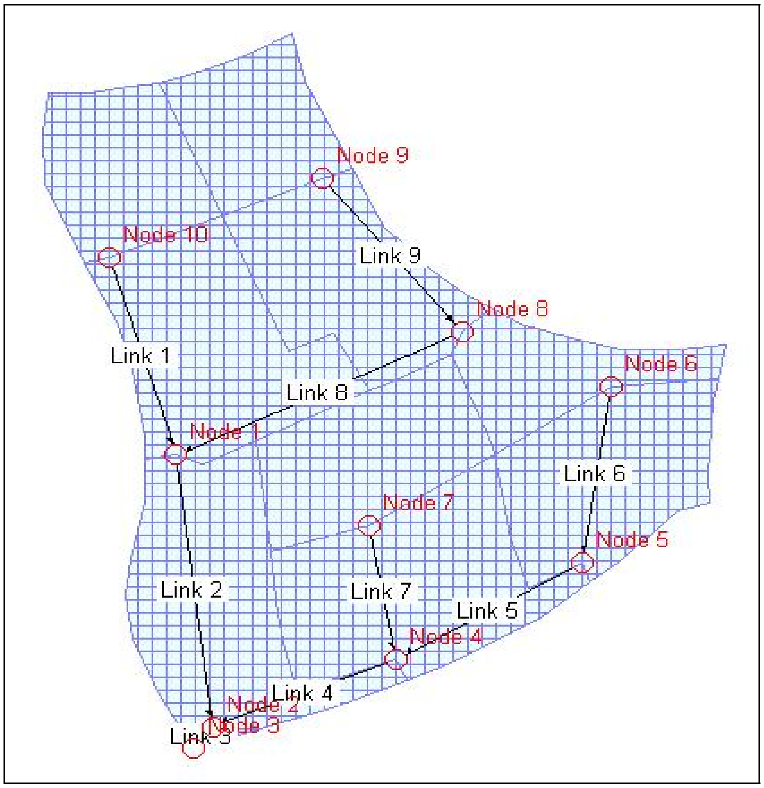


Figure 7-1 Cougar Estates Storm Drain Schematic from xpswmm

3. Double click on the node labeled "Node 9" in Figure 7-1

The Node Data dialog displays the ground elevation, invert elevation, and inflow data as well as the ponding type and initial depth if specified in WMS.

4. Click OK to close the Node Data dialog
5. Double click on the link labeled "Link 9" in Figure 7-1

The Conduit Data dialog displays the pipe data we imported from WMS. On the main window, notice the edit fields for initial flow and initial depth.

6. Click on the button labeled Conduit Profile

   Here we find information about the pipe dimensions and material as well as the invert and ground elevations for the upstream and downstream ends of the pipe.

7. Click OK, then OK again to close both dialogs

Now before we can run SWMM, we need to specify an outfall condition at the downstream outlet of our storm drain system and set up the run time parameters.

8. Double click on the most downstream link in the network (Link 3 in Figure 7-1)
9. Click on the Outfall button

For this exercise, we'll assume a free outfall at the downstream outlet.

10. Click on the Type 1, Free Outfall button
11. In the Outfall Control dialog, select Use normal depth (Yn)

12. *Click OK, OK, then OK to exit all the dialogs*
13. Select ***Configuration | Job Control | Hydraulics***
14. In the Job Control dialog, change the ending day to 1, and the ending hour to 3 **and click** OK **to exit the dialog**
15. Select ***Analyze | Solve*** or click the Solve button 
16. Name the file *"swmmtut.out"* and click Save

7.2 Viewing SWMM Output

1. After the model is done running, select any link and select ***Results | Review Results***

This option will open a plot showing the upstream and downstream water surface elevations at any point throughout the model run and the computed flows and velocities.

2. Click Close and with the same link selected, select ***Results | Dynamic Section Views***
3. Click the Play button 

This option will play an animation of the storm flow as it moves through your selected link. It displays both a profile view and cross sectional view of the flow through the pipe and the flow graph we saw in the previous plot. Feel free to explore the rest of the options under the Results menu or close xpswmm. Leave your WMS project open for the next example.

8 Running EPA SWMM and Viewing Results

This section describes how to export and run an EPA SWMM model. The EPA SWMM model offers many features that can be used to modify your hydrologic analysis, customize your drain system, and analyze the model output. The features covered here represent just a few of the features you might find helpful.

1. Select ***SWMM | Run EPA SWMM…***
2. Name the file "*epaswmmtut.inp"* and click Save

WMS saves an EPA SWMM project file and as long as EPA SWMM is loaded on your computer, it launches EPA SWMM and automatically opens the project we just created.

8.1 Viewing SWMM Model Inputs

1. Your EPA SWMM window should display an image similar to that shown in Figure 8-1.

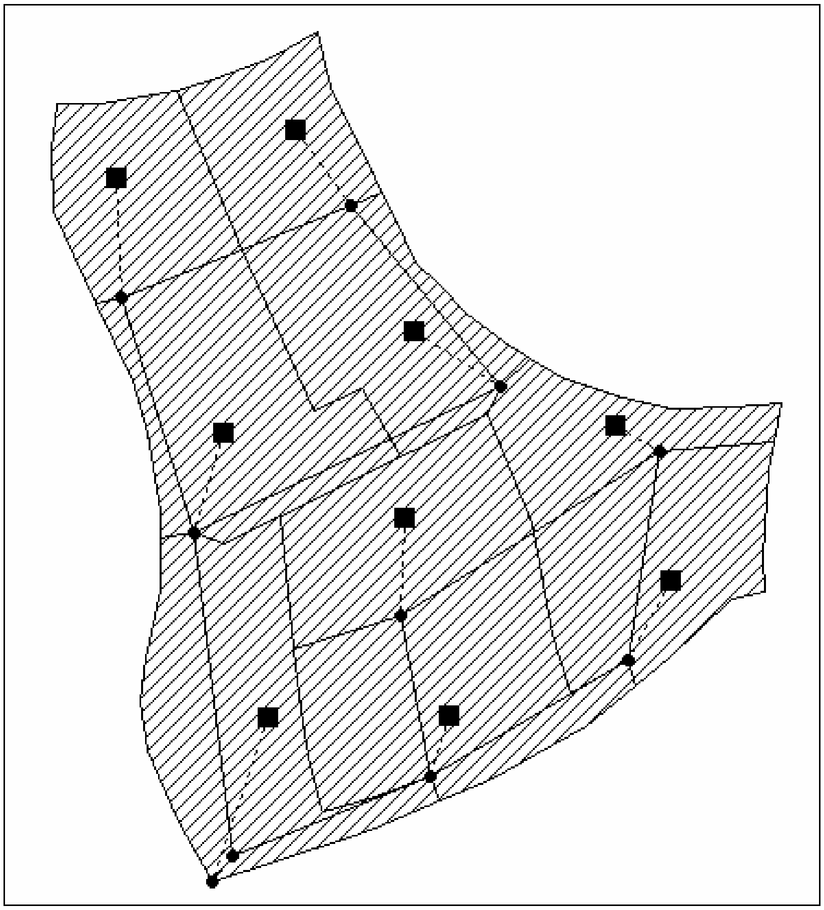


Figure 8-1 Cougar Estates Storm Drain Schematic from EPA SWMM

2. In the SWMM Data Explorer, expand the Hydraulics heading, then the Links heading, and select Conduits
3. In the Conduits window below, select one of the Links
4. Double click the blinking link icon corresponding to the link you selected
5. In the Conduit Link Properties window, notice the link shape, maximum depth, and length which were specified and computed in WMS and exported to EPA SWMM
6. Close the Conduit Link Properties window
7. In the SWMM Data Explorer, expand the Hydraulics heading, then the Nodes heading, and select Junctions
8. In the Junctions window below, select a Node
9. Double click on the blinking node icon corresponding to the node you selected
10. In the Junction Node Properties window, notice the Invert Elevation of the node which was computed in WMS and exported to EPA SWMM
11. Close the Junction Node Properties window

12. In the SWMM Data Explorer, expand the Hydrology heading and select Subcatchments
13. In the Subcatchments window below, select a subcatchment
14. Double click on the blinking subcatchment icon corresponding to the subcatchment you selected
15. In the Subcatchment Properties window, notice that the subcatchment area has already been computed and imported from WMS

Feel free to explore the rest of the model inputs within the EPA SWMM interface.

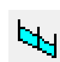
8.2 Running the SWMM Model

1. On the EPA SWMM menu, select ***Project | Run Simulation***

In the Run Status window, you'll notice that the run was successful.

2. On the Run Status window, click OK

8.3 Viewing SWMM Output

1. Select ***Report | Status***
2. Scroll through the report and view the output. Notice that the size of the pipes could be reduced and the flow would still be contained in the pipes without surcharges. Also notice that the Rainfall/Runoff model included with SWMM was not run because the discharges at the nodes have already been defined. If you want to run the Rainfall/Runoff model, you would need to turn this option on in the project options and you would probably want to remove the inflows at each of the nodes since these would be computed using the hydrologic model.
3. Select the most downstream node in your model.
4. Choose the Profile Plot button .
5. Select the "+" button next to the End Node. Now select one of the most upstream nodes and select the "+" button next to the Start Node.
6. Select the "Find Path" button, and then select OK. A plot will appear showing the results of the steady state analysis with the rational flowrates. The plot should look similar to Figure 8-2

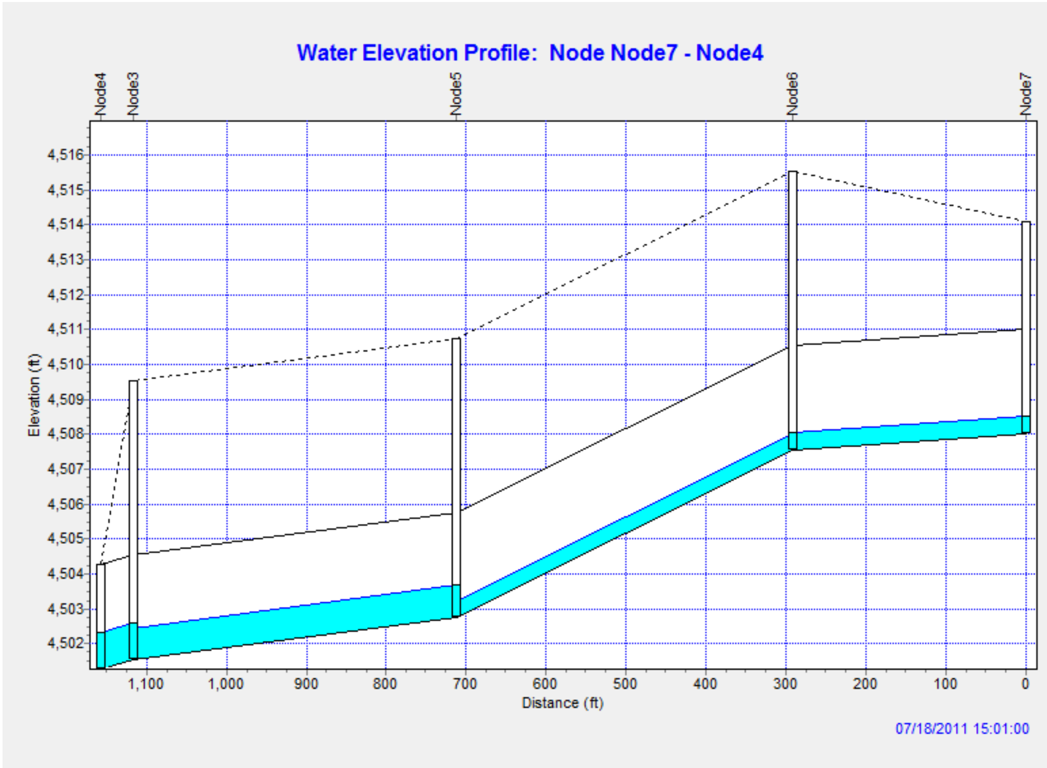


Figure 8-2 Elevation v. Distance Plot from an upstream to a downstream node in your model.

Feel free to explore the rest of the EPA SWMM post processing tools to view the results from the Cougar Estates model run. When you are ready to proceed to the next section, close EPA SWMM (you may want to save your current simulation results if prompted) and leave the WMS window running.

9 Routing Hydrographs in SWMM

In the previous example, we used the Rational Method to compute peak flows which we then fed into our SWMM model. This time, we'll use the Rational Method to compute hydrographs at each outlet. We'll then route these hydrographs through our SWMM model.

9.1 Computing Rational Method Hydrographs

1. In your WMS project, select the Drainage coverage to make it active.
2. Switch to the *Hydrologic Modeling* module .
3. Choose the *Select Outlet* tool .
4. Using the Select Outlet tool, double click on the most downstream outlet (it will most likely be hidden behind the SWMM node icon).
5. In Display, change the Show drop down box to Selected.

6. Click the Compute… button next to the Compute Hydrographs parameter for the selected outlet and click Done in the Rational Method Hydrographs dialog.
7. Click OK to exit the Rational Method dialog.
8. Notice that each basin and outlet icon should show a hydrograph icon next to it as in Figure 9-1.

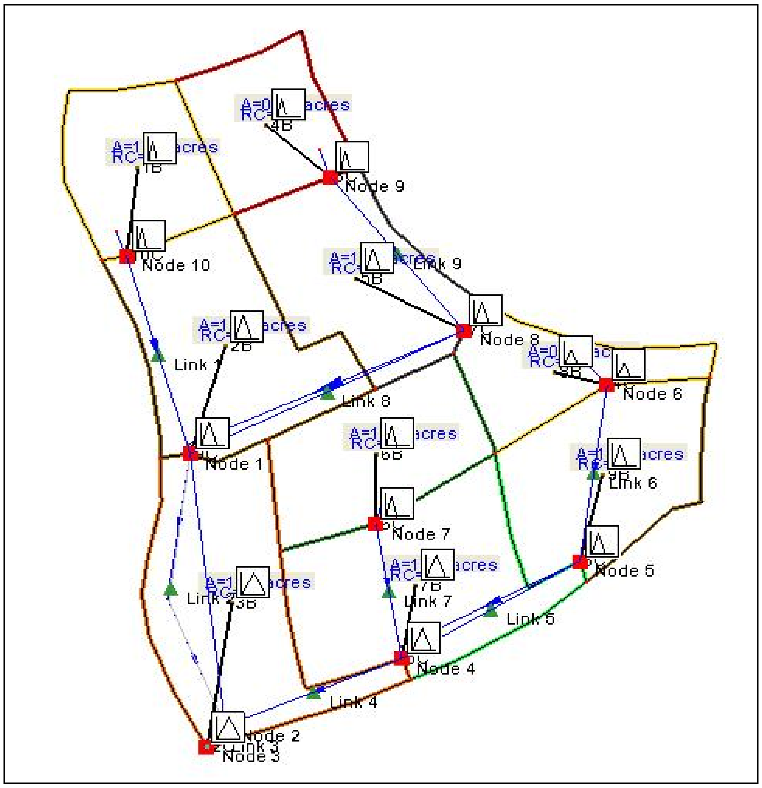


Figure 9-1 Rational Method hydrographs for SWMM

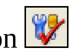
9. In the Project Explorer, select the *SWMM Hydraulic Schematic* tree item.
10. Depending on the model you wish to run, select ***SWMM | Run xpswmm…*** or ***SWMM | Run EPA SWMM….***
11. Name the file "*swmm_hyd.xpx*" (or *swmm_hyd.inp*) and click OK.

9.2 xpSWMM: Running and Viewing Results

1. In the xpswmm interface, double click on any node

Notice that the Inflow edit field under the Constant Inflow category now says 0.0. This is because we have now defined a Time Series Inflow. Notice how the User Inflow button is now checked.

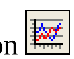
2. Double click the User Inflow button to view the hydrograph we will now be routing through the selected node
3. Click OK, then OK to close both dialogs

4. Double click on the most downstream link in the network (Link 3 in Figure 7-1)
5. Click on the Outfall button
6. Click on the Type 1, Free Outfall button
7. In the Outfall Control dialog, select Use normal depth (Yn)
8. *Click OK, OK, then OK to exit all the dialogs*
9. Select ***Configuration | Job Control | Hydraulics***
10. In the Job Control dialog, change the ending day to 1, and the ending hour to 3 and then click OK to close the dialog
11. Select ***Analyze | Solve*** or click the Solve button 
12. Name the file *"swmm_hyd.out"* and click Save
13. View the results as before and notice the difference between using a constant inflow and a time series inflow

9.3 EPA-SWMM: Running and Viewing Results

1. On the EPA SWMM menu, select ***Project | Run Simulation***

In the Run Status window, you'll notice that the run was successful.

2. On the Run Status window, click OK
3. Select the conduit connecting the most downstream outlet to the junction just upstream
4. Choose the Time Series Plot button 
5. Select any other links where you would like to view output and select the "+" button in the Time Series Plot window to add the plot for these links.
6. After all the desired links have been added, click OK on the Time Series Plot window to view the Flow v. Time plot for the selected storm drains. You could also view the water depths in the storm drains using this same approach. After the plot is displayed, right-click on the plot to view the Graph Options. Select the Horizontal Axis tab and enter a Maximum axis value of 3.0 (hours) and an increment of 0.25 (hours). Click OK to view your final plot.
7. The plot should look similar to Figure 9-2

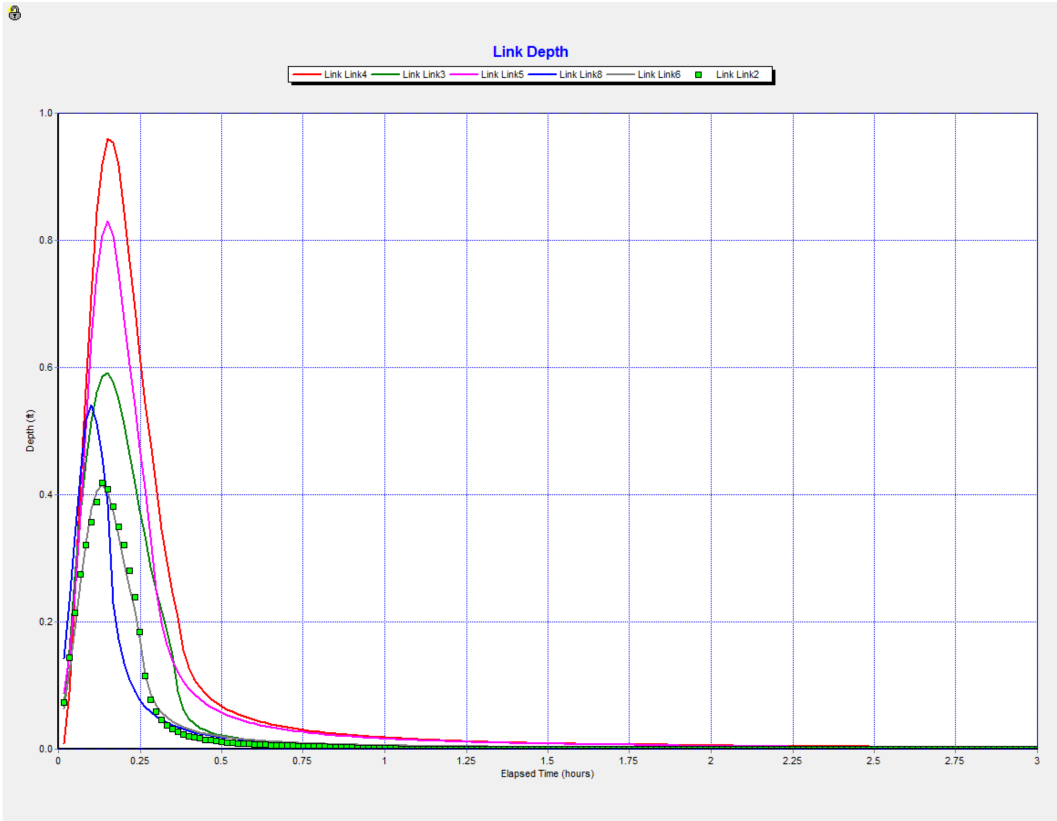


Figure 9-2: Plot of Depth vs. Time for several links in the model.

This concludes the SWMM modeling tutorial. Feel free to explore the WMS interface or the SWMM interface more thoroughly if you wish.